SSNP142 – Modeling of the rupture in the presence of under pressures and slip of a stopping with the elements of joint

Summary:

In this test one models the setting in water of a stopping weight. Various modelings take into account the propagation of under pressures, the rupture or the slip (friction) of stopping on the level of its foundation. In the whole case modeling is carried out using the elements of joint (`XXX_JOINT or XXX_JOINT_HYME`) and of the laws `JOINT_MECA_RUPT` and `JOINT_MECA_FROT`.

Modeling a: Stopping $2D$, `JOINT_MECA_RUPT` and under imposed pressures (`PLAN_JOINT)` linear
Modeling b: Stopping $3D$ [1], `JOINT_MECA_RUPT` and under imposed pressures (`3D_JOINT)` linear
Modeling C: Stopping $2D$, `JOINT_MECA_FROT` and under imposed pressures (`PLAN_JOINT)` linear
Modeling D: Stopping $3D$, `JOINT_MECA_FROT` and under imposed pressures (`3D_JOINT)` linear
Modeling E: Stopping $2D$, `JOINT_MECA_RUPT,` propagation of under pressures (`PLAN_JOINT_HYME`)
Modeling F: Stopping $3D$ [1], `JOINT_MECA_RUPT,` propagation of under pressures (`3D_JOINT_HYME`)
Modeling G: Stopping $2D$, `JOINT_MECA_FROT,` propagation of under pressures (`PLAN_JOINT_HYME`)
Modeling H: Stopping $3D$ [1],`JOINT_MECA_FROT,` propagation of under pressures (`3D_JOINT_HYME`)
Modeling I: Stopping $2D$, `JOINT_MECA_RUPT` and under imposed pressures (`PLAN_JOINT)` fortysomething
Modeling J: Stopping $2D$, `JOINT_MECA_FROT` and under imposed pressures (`PLAN_JOINT)` fortysomething
Modeling K: Stopping $2D$, `JOINT_MECA_RUPT` test of stability after rupture of modeling A

---

[1] Grid $3D$ is obtained by the extrusion of the model $2D$.

Results of modelings with the law `JOINT_MECA_RUPT` (A, B, E, F and I) are validated by comparison with those obtained with the computer code *GEFDYN,* currently used by the Hydraulic Center of Engineering of EDF.

1 Problem of reference

1.1 Geometry

One considers a stopping in the shape of trapezoid of great base $5m$, of small base $1.5m$ and height $10m$. This last is posed in the center of a rectangular foundation of $15m$ of length and $5m$ of top (see figure 1.1). For modeling $3\mathrm{D}$ dimensions are identical in the plan (x, y) and the unit is extruded of $1m$ in the direction z (of $10cm$ for modeling F).

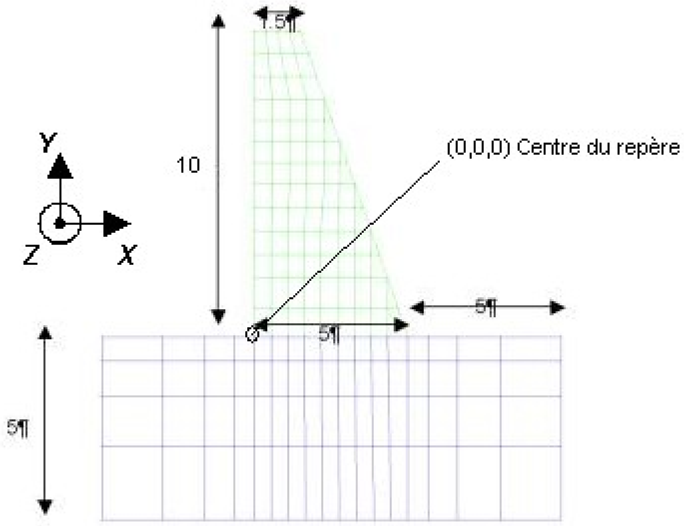


Figure 1.1: Geometry of the stopping and the foundation

1.2 Properties of material

The values of the mechanical parameters of the stopping (Young modulus, Poisson's ratio, voluminal density of the stopping and water) are in the following way selected:

$$E=3.10^2 Pa \quad \nu=0.25 \quad \rho_b=2400\, kg/m^3 \quad \rho_e=1000\, kg/m^3$$

1.2.1 Modelings A, B, I, K: law `JOINT_MECA_RUPT`

These modelings test the mechanical behavior with under imposed pressures. For the joint one takes the normal stiffness equal to the tangential stiffness. There is no tensile strength. The coupling between the normal opening and the tangential stiffness is selected in order to have the worthless tangential slope as soon as the joint reaches the threshold of complete normal damage. The slope of softening in rupture is five times stiffer than the normal slope of loading (see document [`R7.01.25`]).

$$K_N=K_T=10^{12} Pa/m \qquad \sigma_{max}=0\, Pa$$

$$\alpha=1 \qquad \mathrm{pena\_rupt}=0.2$$

(NB: the values “tests” provided by the CIH do not correspond to any material in particular)

1.2.2 Modelings C, D and J: law `JOINT_MECA_FROT`

The joint is modelled by an elastoplastic law of standard friction Mohr-Coulomb (see [`R7.01.25`]), which depends on five parameters. Two elastic parameters: one takes the tangential stiffness equal to the double of the normal stiffness. Two parameters of the law Mohr-Coulomb: adherence and the coefficient of friction. More one parameter of regularization of the tangent matrix in slip.

$K_N = 10^{12}\, Pa/m$ $\quad$ $\text{adhésion} = 1\, kPa$ $\quad$ $\text{pena\_tang} = 0.1 \cdot K_T$

$K_T = 2 * K_N$ $\quad$ $\mu = 0.35$

1.2.3 Modelings E and F: law `JOINT_MECA_RUPT`

These modelings test the hydro-mechanical behavior with the propagation of under pressures. For the joint one takes the normal stiffness equal to the tangential stiffness. The tensile strength has a weak value in order to facilitate the convergence of calculation. The coupling between the normal opening and the tangential stiffness is selected in order to have the worthless tangential slope as soon as the joint reaches the threshold of complete normal damage. The slope of softening in rupture is five times stiffer than the normal slope of loading (see document [`R7.01.25`]). The hydraulic parameters are those of water.

$K_N = K_T = 10^{12}\, Pa/m$ $\quad$ $\sigma_{max} = 100\, Pa$ $\quad$ $\rho_{\text{eau}} = 1000\, kg/m^3$

$\alpha = 1$ $\quad$ $\text{pena\_rupt} = 0.2$ $\quad$ $visc_{\text{eau}} = 10^{-3}\, Pa \cdot s$

(NB: the values “tests” provided by the CIH do not correspond to any material in particular)

1.2.4 Modelings G and H: law `JOINT_MECA_FROT`

These modelings test the hydro-mechanical behavior with the propagation of under pressures. The joint is modelled by an elastoplastic law of standard friction Mohr-Coulomb (see [`R7.01.25`]), which depends on five parameters. Two elastic parameters: one takes the tangential stiffness equal to the double of the normal stiffness. Two parameters of the law Mohr-Coulomb: adherence and the coefficient of friction. More one parameter of regularization of the tangent matrix in slip. The hydraulic parameters are those of water.

$K_N = K_T / 2 = 10^{12}\, Pa/m$ $\quad$ $\text{adhésion} = 1\, kPa$ $\quad$ $\rho_{\text{eau}} = 1000\, kg/m^3$

$\text{pena\_tang} = 0.1 \cdot K_T$ $\quad$ $\mu = 0.4$ $\quad$ $visc_{\text{eau}} = 10^{-3}\, Pa \cdot s$

1.3 Boundary conditions and loadings

1.3.1 Mechanical loadings

The interface between the stopping and the foundation is modelled by finite elements of joint. The lower parts of the foundation are embedded. The stopping is subjected to the gravitational force and one gradually fills out of water the upstream part of the stopping (left on the figure). That amounts applying a pressure distributed P_{amont} on the face $(x=0, y \in [0,10])$ that one expresses according to y :

$$P_{amont} = \rho_e\, g (n_e - y)$$

where g indicate the acceleration of gravity ρ_e density of water and n_e the water level.

1.3.2 Under pressure

For pure mechanical modelings (A, B, C, D, I, J) one imposes a profile of under pressures given.
To model the water propagation in the crack under the stopping (in $y=0$) one takes into account a pressure of fluid P_{fluide} linear. This one is worth $\rho_e g n_e$ upstream $(x=0)$ and is worthless downstream $(x=5)$.
In modelings E and F one takes into account the propagation of under pressures (one imposes only the pressure upstream and the pressure downstream), it is the calculation which will provide the profile of under pressures.

Notice : In a preoccupation with a robustness of calculations one uses `AFFE_CHAR_CINE` for the boundary conditions on the joints (modelings E and F). Moreover one imposes the keyword `NPREC=-1` in the part solvor of `STAT_NON_LINE`.

2 Reference solution

For modelings with the law `JOINT_MECA_RUPT` (A, B, E, F and I), the reference solution is given by the computer code GEFDYN [bib1]. Modelings C, D, G, H and J are used as test of robustness and nonregression of the model of friction.

2.1 Uncertainties on the solution

For GEFDYN, the relative precision is fixed at 1E-2 and the grid is of five to ten times coarser, which explains a variation (and not an error) about 10% between modelings GEFDYN and Code_Aster.

2.2 Bibliographical references

[1] GEFDYN, Geomechanical Elements Finis Dynamiques. Analysis coupled in Mechanics/Hydraulic/Thermal of the nonlinear behavior of géomatériaux quasi static and/or dynamic 2D/3D. MSSMat laboratory, Central School Paris.

3 Modeling A

3.1 Characteristics of modeling

Simulation is carried out with modeling `PLAN_JOINT`. The elements are of type `TRIA3` for the stopping and the foundation and of type `QUAD4` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT`, the associated material bears the same name. The surface elements are elastic.

3.2 Characteristics of the grid

A linear grid is carried out:
Voluminal elements (stopping and foundation): 2264 `TRIA3`
Elements of joint: 50 `QUAD4`

3.3 Sizes tested and results

One compares the results with those of GEFDYN. One notes δ_n (`V7`) the normal opening of the joints and σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

Size tested	***GEFDYN***	***Tolerance***
δ_n in $x=0\,m$	4.01D-7	7%
δ_n EN $x=5\,m$	-4.25D-7	5%
σ_n EN $x=0\,m$	-8.83D+04	1%
σ_n EN $x=5\,m$	-4.250D+05	5%

Tests of nonregression are also carried out to make sure of the stability of the data-processing developments, they are not presented here. One represents the deformation of the stopping and the opening of the crack on figure 3.3 below. To allow a good visualization, a multiplicative factor is applied.

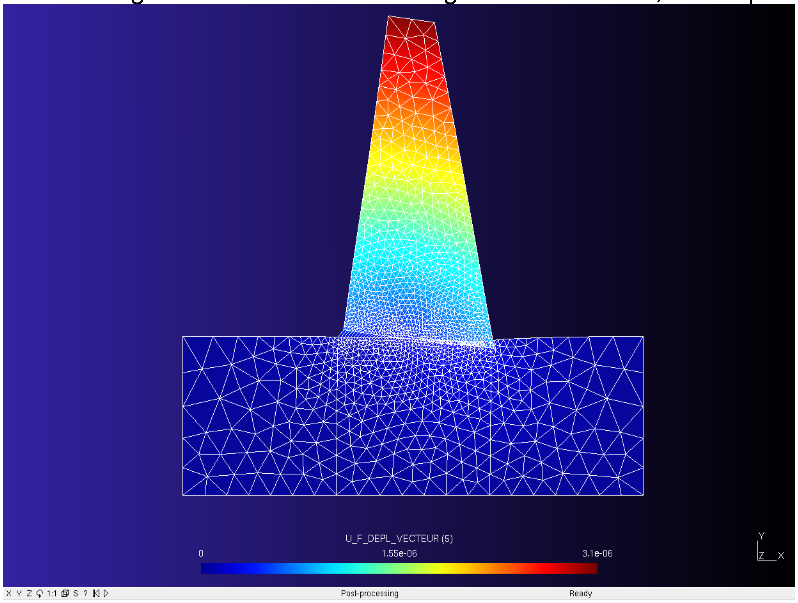


Figure 3.3: Deformation of the stopping and opening of the crack

4 Modeling B

4.1 Characteristics of modeling

Simulation is carried out with modeling `3D_JOINT`. The elements are of type `TETRA4` for the stopping and the foundation and of type `PENTA6` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT`, the associated material bears the same name. The voluminal elements are elastic.

4.2 Characteristics of the grid

A linear grid is carried out:
Éléments voluminal (stopping and foundation): 37583 `TETRA4`
Éléments of joint: 1146 `PENTA6`

4.3 Sizes tested and results

One compares the results with those of GEFDYN. One notes δ_n (`V7`) the normal opening of the joints and σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

Size tested	***GEFDYN***	***Tolerance***
δ_n in $x=0m$	4.01D-7	5%
δ_n EN $x=0m$	-4.25D-7	3%
σ_n EN $x=5m$	-8.83D+04	1%
σ_n EN $x=5m$	-4.250D+05	3%

Tests of nonregression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

5 Modeling C

5.1 Characteristics of modeling

Simulation is carried out with modeling `PLAN_JOINT`. The grid is regular. The elements are of type QUAD4 and TIRA3 for the stopping and the foundation and of type `QUAD4` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_FROT`, the associated material bears the same name. The surface elements are elastic.

5.2 Characteristics of the grid

A linear grid is carried out:
Voluminal elements (stopping and foundation): 34 `TRIA3 and 860 QUAD4`
Elements of joint: 50 `QUAD4`

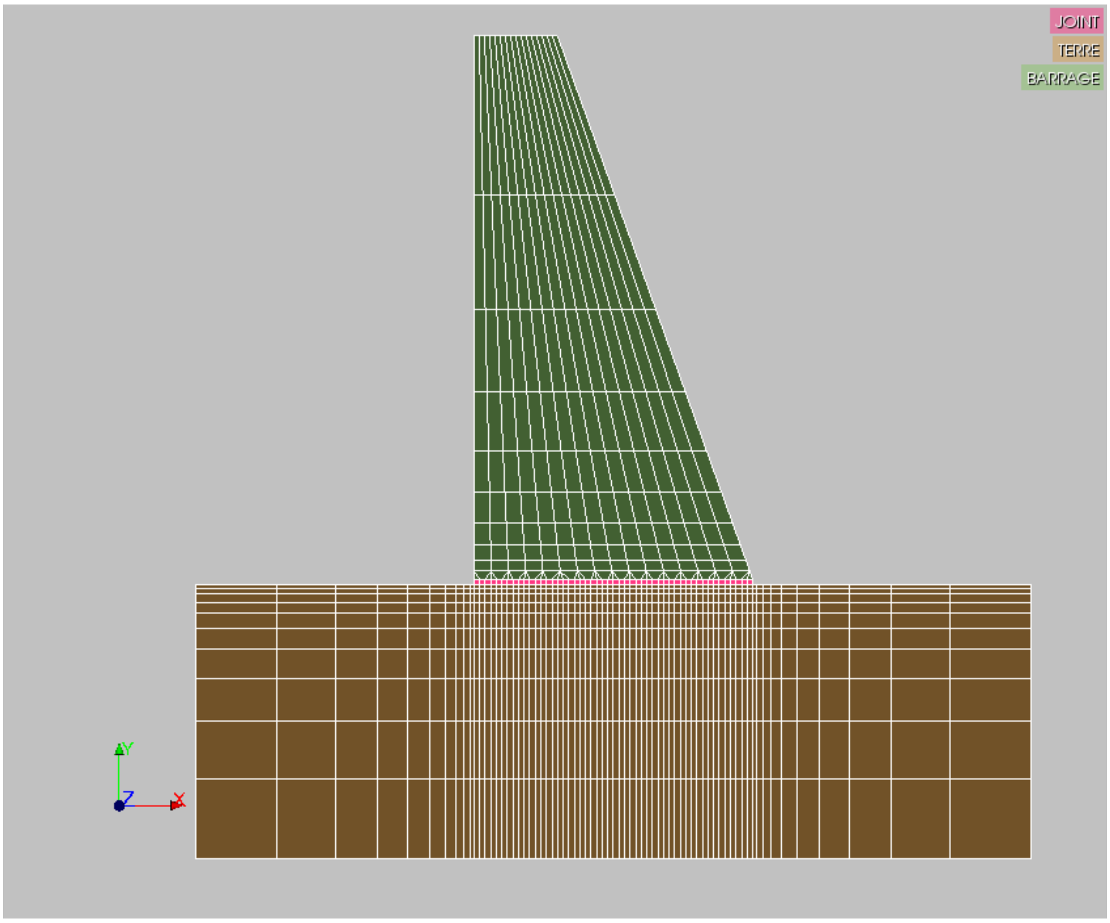


Figure 5.2-1: Regular grid of modeling C

5.3 Sizes tested and results

Tests of nonregression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via `V7 or DEPL`) the normal opening of the joints, σ_n (`SIGN`) the normal constraint and then pressure of fluid imposed (variable V18). The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

6 Modeling D

6.1 Characteristics of modeling

Simulation is carried out with modeling `3D_JOINT`. The elements are of type `TETRA4` for the stopping and the foundation and of type `PENTA6` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_FROT`, the associated material bears the same name. The voluminal elements are elastic.

6.2 Characteristics of the grid

A linear grid is carried out:
Éléments voluminal (stopping and foundation): 37583 `TETRA4`
Éléments of joint: 1146 `PENTA6`

6.3 Sizes tested and results

Tests of nonregression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via `V7 or DEPL`) the normal opening of the joints, σ_n (`SIGN`) the normal constraint, then pressure of fluid imposed (variable V18). The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

As illustration one presents the visualization of constraints (on the left) and the part in slip (on the right) of the stopping on figure 6.3.

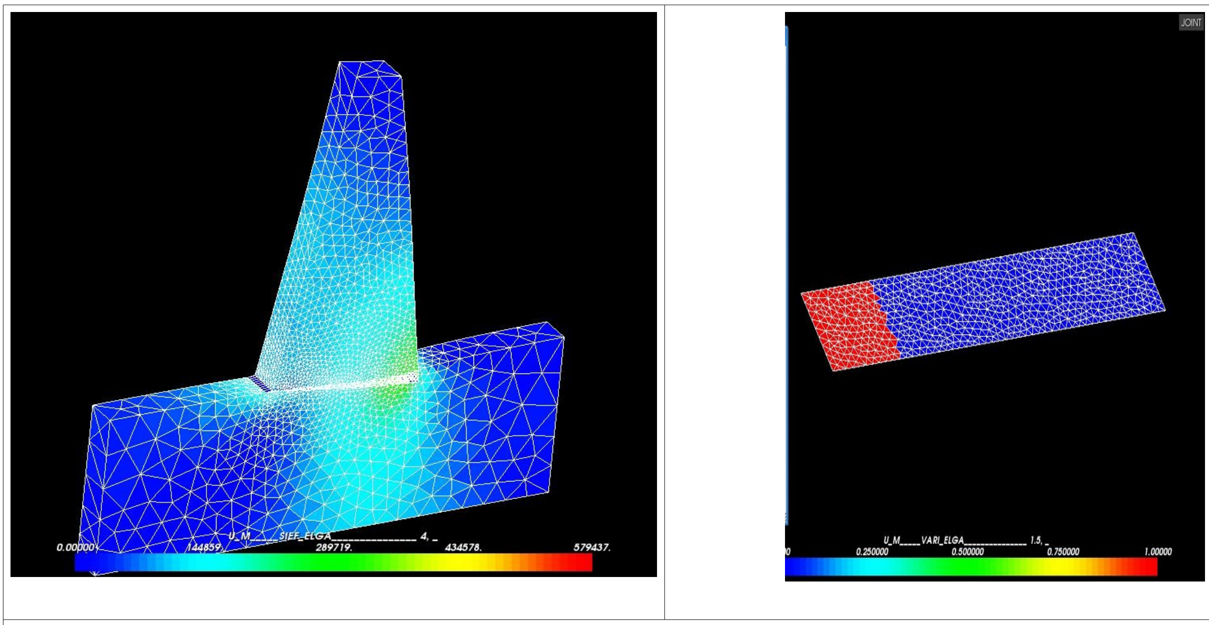


Figure 6.3: Constraints on the stopping and the part of the slip plane (in red)

7 Modeling E

7.1 Characteristics of modeling

Simulation is carried out with modeling `PLAN_JOINT_HYME`. The elements are of type `TRIA6` for the stopping and the foundation and of type `QUAD8` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT,` the associated material bears the same name. The surface elements are elastic.

7.2 Characteristics of the grid

A quadratic grid is carried out:
Voluminal elements (stopping and foundation): 3626 `TRIA6`
Elements of joint: 100 `QUAD8`

7.3 Sizes tested and results

One compares the results with those of GEFDYN. One notes δ_n (`V7`) the normal opening of the joints and σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

Size tested	***GEFDYN***	***Tolerance***
δ_n in $x=0\,m$	1.42E-06	8%
σ_n EN $x=0\,m$	-88290	1%
δ_n EN $x=5\,m$	-5.84E-07	15%
σ_n EN $x=5\,m$	-5.86E+05	15%

Tests of nonregression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

8 Modeling F

8.1 Characteristics of modeling

Simulation is carried out with modeling `3D_JOINT_HYME`. The elements are of type `TETRA10` for the stopping and the foundation and of type `PENTA15` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT`, the associated material bears the same name. The voluminal elements are elastic. The component `DZ` is put at zero on the edges of the section of stopping in order to reproduce the behavior in plans displacements.

8.2 Characteristics of the grid

A linear grid is carried out:
Éléments voluminal (stopping and foundation): 7435 `TETRA10`
Éléments of joint: 200 `PENTA15`

8.3 Sizes tested and results

One compares the results with those of GEFDYN. One notes δ_n (`V7`) the normal opening of the joints and σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

Size tested	***GEFDYN***	***Tolerance***
δ_n in $x=0m$	1.44E-06	7%
σ_n EN $x=0m$	-88290	1%
δ_n EN $x=5m$	-5.89E-07	15%
σ_n EN $x=5m$	-5.89E+05	15%

Tests of nonregression are also carried out to make sure of the stability of the data-processing developments, they are not presented here. One carries out in particular tests on the variable interns 18 which makes it possible to know the value of the pressure of fluid interpolated at the points of Gauss.

9 Modeling G

9.1 Characteristics of modeling

Simulation is carried out with modeling `PLAN_JOINT_HYME`. The elements are of type `TRIA6` for the stopping and the foundation and of type `QUAD8` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_FROT`, the associated material bears the same name. The surface elements are elastic.

9.2 Characteristics of the grid

A quadratic grid is carried out:
Voluminal elements (stopping and foundation): 3626 `TRIA6`
Elements of joint: 100 `QUAD8`

9.3 Sizes tested and results

Tests of nonregression are carried out to make sure of the stability of the data-processing developments. One test δ_n (obtained via `V7 or DEPL`) the normal opening of the joints, σ_n (`SIGN`) the normal constraint . The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`). One carries out in particular tests on the variable interns 18 which makes it possible to know the value of the pressure of fluid interpolated at the points of Gauss.

10 Modeling H

10.1 Characteristics of modeling

Simulation is carried out with modeling `3D_JOINT_HYME`. The elements are of type `TETRA10` for the stopping and the foundation and of type `PENTA15` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_FROT`, the associated material bears the same name. The voluminal elements are elastic. The component `DZ` is put at zero on the edges of the section of stopping in order to reproduce the behavior in plans displacements.

10.2 Characteristics of the grid

A linear grid is carried out:
Éléments voluminal (stopping and foundation): 7435 `TETRA10`
Éléments of joint: 200 `PENTA15`

10.3 Sizes tested and results

Tests of nonregression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via `V7 or DEPL`) the normal opening of the joints, σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`). One carries out in particular tests on the variable interns 18 which makes it possible to know the value of the pressure of fluid interpolated at the points of Gauss.

11 Modeling I

11.1 Characteristics of modeling

It is equivalent modeling A into quadratic. Simulation is carried out with modeling `PLAN_JOINT`. The elements are of type `TRIA6` for the stopping and the foundation and of type `QUAD8` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT`, the associated material bears the same name. The surface elements are elastic.

11.2 Characteristics of the grid

A quadratic grid is carried out:
Voluminal elements (stopping and foundation): 2264 `TRIA6`
Elements of joint: 50 `QUAD8`

11.3 Sizes tested and results

One compares the results with those of GEFDYN. One notes δ_n (`V7`) the normal opening of the joints and σ_n (`SIGN`) the normal constraint. The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

Size tested	***GEFDYN***	***Tolerance***
δ_n in $x=0m$	4.01D-7	7%
δ_n EN $x=5m$	-4.25D-7	5%
σ_n EN $x=0m$	-8.83D+04	1%
σ_n EN $x=5m$	-4.250D+05	5%

Tests of nonregression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

12 Modeling J

12.1 Characteristics of modeling

It is equivalent quadratic of modeling B. simulation is carried out with modeling `PLAN_JOINT`. The grid is regular. The elements are of type QUAD8 and TIRA6 for the stopping and the foundation and of type `QUAD8` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_FROT`, the associated material bears the same name. The surface elements are elastic.

12.2 Characteristics of the grid

A linear grid is carried out:
Voluminal elements (stopping and foundation): 34 `TRIA6 and 860 QUAD8`
Elements of joint: 50 `QUAD8`

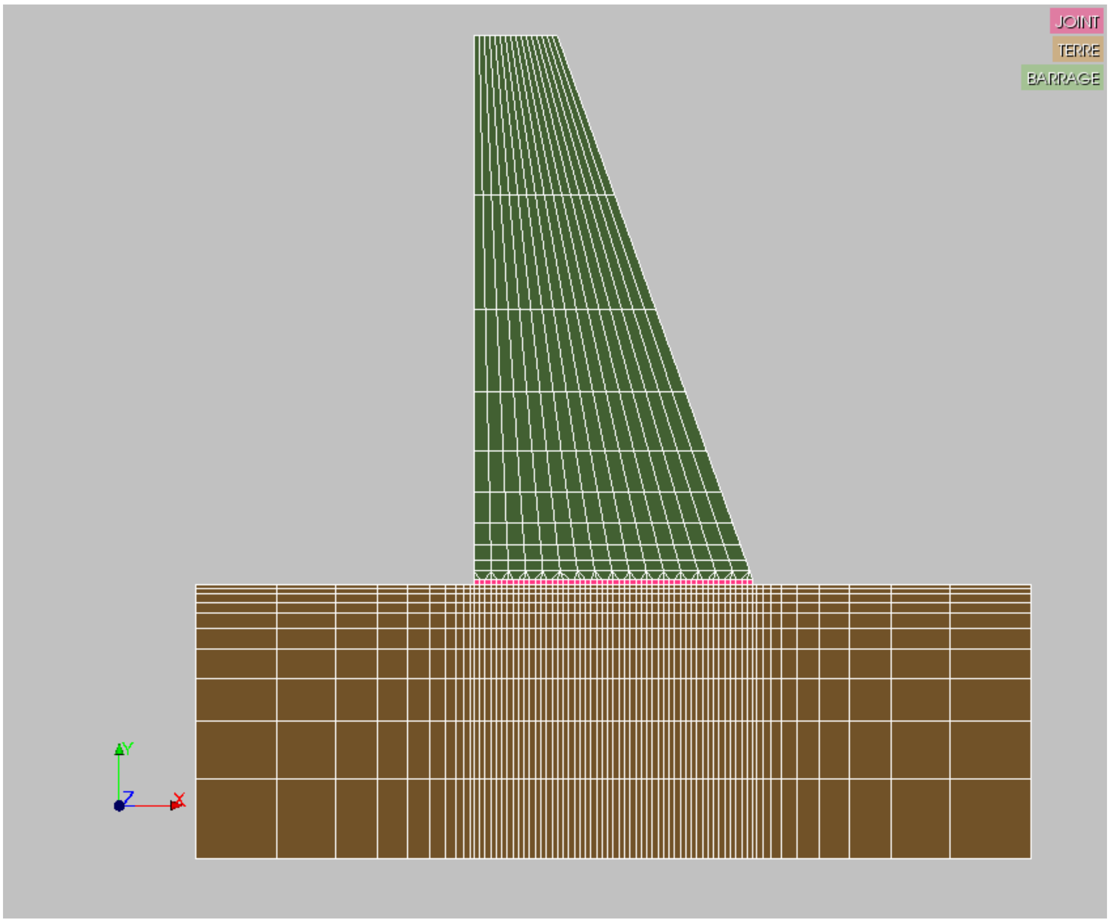


Figure 12.2-1: Regular grid of modeling C

12.3 Sizes tested and results

Tests of nonregression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via `V7 or DEPL`) the normal opening of the joints, σ_n (`SIGN`) the normal constraint and then pressure of fluid imposed (variable V18). The values tested result from extrapolations to the nodes (`VARI_NOEU` and `SIEF_NOEU`).

13 Modeling K

In this case test one checks the stability of the answer of stopping after he reached the critical water level (calculation RDM). It is, that simulations have a very delicate convergence currently there and it is extremely difficult to know if one really reached the critical load (see Fig 13.1). In order to help post-critical convergence one uses the technique of piloting by `DDL_IMPO` on the upper lip of the joint upstream of the stopping. In order to simplify analysis the pressure is applied in a homogeneous way to the face upstream of the work.

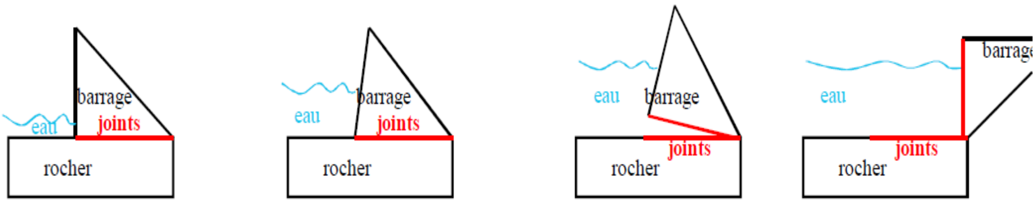


Figure 13.1: Deformation of the stopping and possible opening of the crack

13.1 Characteristics of modeling

Simulation is carried out with modeling `PLAN_JOINT`. The elements are of type `TRIA3` for the stopping and the foundation and of type `QUAD4` for the elements of joint. The corresponding law of behavior is `JOINT_MECA_RUPT`, the associated material bears the same name. The surface elements are elastic.

13.2 Characteristics of the grid

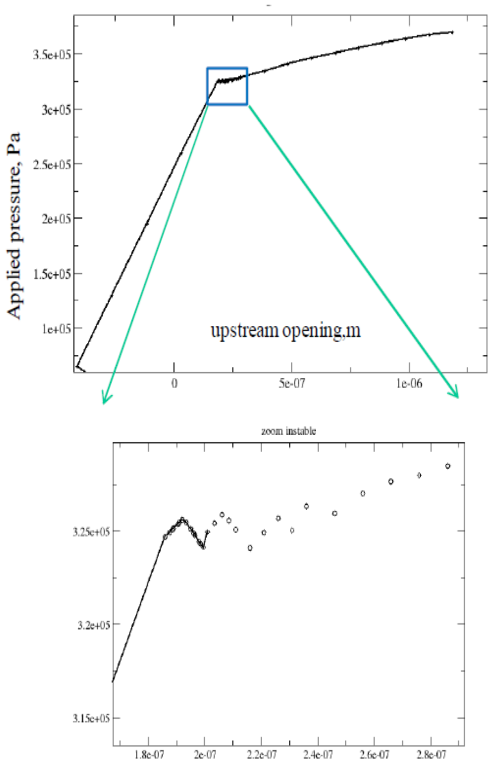
A linear grid is carried out:
Voluminal elements (stopping and foundation): 2264 `TRIA3`
Elements of joint: 50 `QUAD4`

13.3 Sizes tested and results

The curved pressure is analyzed -- opening of joint. The absence of snap-back confirms the stability of the work even after the theoretical critical load (calculation RDM). The difficulty of convergence is then related to the brutal rupture several meshs of joint on the level of this critical loading.

One compares the results of the curved pressure – opening of joint.

Tests of nonregression are carried out to make sure of the stability of the data-processing developments, they are not presented here.



14 Summary of the results

Modelings A, B, E, F and I provide results in conformity with GEFDYN. That makes it possible to validate the law of behavior `JOINT_MECA_RUPT` at the same time on a mechanical and hydro-mechanical level, that also validates the taking into account of the pressure of fluid on the lips of the crack using the keyword `PRES_FLUIDE`. Modelings C, D, G, H and J validate the robustness of the version full-implicit of the law of friction `JOINT_MECA_FROT.` In modeling K one test at the same time the stability of stopping after the critical load, as well as the method of piloting of loading by ddl imposed.